

Modeling of wind speed in the atmospheric boundary layer in the presence of hill like obstacles

N. AMAHJOUR^a, A. KHAMLICH^b

a. Systems of Communication and Detection Lab., Abdelmalek Essaadi University, Morocco

Email: narjisse.amahjour@gmail.com

b. Systems of Communication and Detection Lab., Abdelmalek Essaadi University, Morocco

Email: khamlichi7@yahoo.es

Résumé :

La vitesse du vent dans la couche limite atmosphérique dépend considérablement de la rugosité de la surface du sol et de la température. La présence d'un obstacle macroscopique sur un terrain plat modifie les caractéristiques d'écoulement d'air à cet endroit. Dans ce travail, l'écoulement turbulent de l'air induit par un obstacle de type colline a été étudié en utilisant une approximation bidimensionnelle du problème. Des conditions atmosphériques neutres ont été considérées. La modélisation par les équations de Navier-Stokes couplées au modèle de la turbulence $k-\varepsilon$ a été choisie. Une loi de paroi améliorée a été appliquée sur le sol. Le logiciel COMSOL Multiphysics basé sur la méthode des éléments finis a été utilisé pour la simulation. La convergence du modèle en fonction de la taille du maillage et de l'emplacement de la frontière du champ de calcul a été étudiée. La validation du calcul CFD a été conduite par comparaison avec FLUENT et des données expérimentales issues du test à la soufflerie RUSHIL H3. Les résultats obtenus pour un site présentant un obstacle de type colline ont montré que le profil de la vitesse du vent est largement affecté par la présence d'un obstacle topographique. Il est possible alors de ré-estimer de façon plus fiable l'énergie extraite du vent par une éolienne standard.

Abstract:

Wind velocity in the atmospheric boundary layer depends hugely on the soil surface roughness and temperature. The presence of a macroscopic obstacle on a flat ground modifies the characteristics of air flow in that site. In this work, turbulent flow of air as induced by a hill like obstacle was analyzed by using a two dimensional approximation of the problem. Neutral atmospheric conditions were assumed. Modelling was achieved by means of Navier-Stokes equations coupled to the $k-\varepsilon$ a turbulence. Enhanced wall law was applied at the ground. The code COMSOL Multiphysics which is based on the finite element method was used for simulations. Convergence of the model as function of the mesh grid and the computational domain was assessed. Validation of the CFD modelling was performed by comparison with FLUENT and experimental results of the wind tunnel test RUSHIL H3. The obtained results for a site with hill like obstruction have shown that the wind speed profile is largely affected by the presence of a topographic obstacle. It is then possible to re-estimate more reliably the energy extracted from wind by a standard wind turbine.

Key words: Wind flow, boundary layer atmospheric, CFD, boundary conditions, wall law, wind turbine.

1 Introduction

Flow over complex terrain has been a subject of interest in many research areas such as determination of air pollution zones, predictions of smoke movement from forest fires, wind energy assessment and optimization of wind energy conversion systems installation. In addition to wind speed-up over hill crest, turbulence effects due to topography and flow separation are important features that have to be considered for adequate wind energy assessment. These effects can be studied by using various approaches: theoretical analysis, numerical modeling and wind tunnel experimentation. Focus is in the following on Computational Fluid Dynamics (CFD) based modeling [1]. This approach consists of solving the differential equations governing fluid flow by using advanced numerical discretization techniques within the context of approximate physical modeling of turbulence. In principle, with sufficient grid refinement, the solution of the discretized governing equations yields a flow that constitutes a reasonable representation of reality, according to the underlying assumptions used to derive the model equations.

The application of CFD to study air flow over a hill in the Atmospheric Boundary Layer (ABL) has been the subject of broad research activity [2,8]. Validation of CFD based modeling has been an essential aspect of this research and several comparative studies between CFD results and wind tunnel data or field measurements have been performed [3-8]. In most of these publications the dispersion process in the wind tunnel and field measurements was judged to be slightly different from that predicted by the Reynolds Averaged Navier-Stokes (RANS) solution as obtained by CFD simulation. This was pointed out to be the main cause for the relative discrepancies existing between experimental data and numerical results. However, the CFD based approach is widely recognized today to be suitable, from an engineering point of view, to get a thorough insight in the complex mechanics governing fluid flowing problems. It enables, by avoiding the high cost needed in the framework of experimental based methods, to get a reasonable accuracy of the output variables with realistic computational time.

Many software packages are now available for users in order to perform CFD analysis of fluid flows. They generally share the advantage of being very compelling, non-intrusive and comprising powerful visualization capabilities. They are commonly based on finite volume technique of finite element method [9]. Here, choice is made of COMSOL Multiphysics software which is based on the finite element method [10].

In this work, a two-dimensional approximation of the fluid flow problem over a hill in the ABL is assumed. The infinite atmospheric field is modeled by a finite domain around the obstacle and which has a rectangular form. The boundary condition at the ground surface is fixed by a modified wall law based on near wall treatment. The other boundary conditions include a logarithmic velocity profile applied at the vertical input section, a uniform speed applied at the upper edge of the rectangular domain which is chosen to have a constant altitude and finally a uniform pressure at the vertical outlet section of the domain.

After assessing convergence of the finite element model through modifying the rectangle area as well as the mesh size, inspection of adequacy of the mesh with the wall law is performed. The influence of the obstacle geometry on the wind speed distribution is then analyzed.

Validation of the CFD based simulation of atmospheric air flow as mentioned previously is performed in the following by comparison of the obtained numerical results with the corresponding wind tunnel experimental data for the "Russian Hill" (RUSHIL) wind-tunnel study [11] and also with available simulation results provided by the CFD platform FLUENT. The obtained simulation results can be used finally to re-estimate the profiles of wind speed at various locations that are considered as potential sites for wind turbine implementation.

2 Mathematical model and numerical techniques

2.1 Fundamental equations

The choice of a specific CFD model depends basically on the nature of the physical process to be simulated. It is reliant also on the objectives pursued through the numerical study and the available computational resources. The mathematical model should be enough detailed to get adequate representation at the targeted turbulence scale, but without making the computations too cumbersome. In practice, the use of a sophisticated model makes it too difficult to develop and implement an efficient numerical algorithm to perform simulation with reasonable cost. In many cases, the desired information can be obtained more effectively by using a simplified modeling of the problem that exploits some a priori knowledge of the flow pattern or incorporates empirical correlations that are supported by theoretical or experimental studies. A compilation of fundamental, phenomenological, and empirical models is available for a number of complicated problems, such as turbulent air flow in the atmospheric layer near the ground.

In the present work, the approach used is purely theoretical and CFD modeling is implemented by using COMSOL Multiphysics software package [10].

a- Reynolds-Averaged (RANS) Navier-Stokes equations

Considering the steady state regime of an incompressible Navier-Stokes flowing fluid, the time averaged RANS equations write [1]:

$$\vec{u} \cdot \vec{\nabla} \vec{u} = -\vec{\nabla} \bar{p} + \vec{\nabla} \cdot \left[(\nu + \nu_T) (\vec{\nabla} \vec{u} + (\nabla \vec{u})^T) \right], \quad \vec{\nabla} \cdot \vec{u} = 0 \quad (1)$$

where \vec{u} is the mean velocity vector, p the mean pressure, ν the kinematics viscosity that depends only on the physical properties of the fluid and ν_T is the turbulent eddy viscosity which is supposed to emulate of unresolved velocity fluctuations \vec{u}' .

b- k- ε two-equation turbulence model

The k- ε turbulence model is the most common model used in CFD to simulate mean flow characteristics for turbulent flow conditions. It is a two transport equations based model which gives a general description of turbulence.

The exact k- ε equations contain many unknown terms. For a much more practical approach, the standard k- ε turbulence model which was introduced by Launder and Spalding [12] is usually used. This model is based on best understanding of the relevant processes, thus minimizing unknowns and presenting a set of equations which can be applied to a large number of turbulent applications. The standard k- ε turbulence model equations write:

$$k \vec{u} = \vec{\nabla} \cdot \left[\left(\nu + \frac{\nu_T}{\sigma_K} \right) \vec{\nabla} k \right] + P_K - \varepsilon \quad (2)$$

$$\vec{\nabla} \cdot (\varepsilon \vec{u}) = \vec{\nabla} \cdot \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \vec{\nabla} \varepsilon \right] + C_{\varepsilon 1} \frac{\varepsilon}{k} P_K - C_{\varepsilon 2} \frac{\varepsilon^2}{k} \quad (3)$$

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (4)$$

$$P_K = \nu_t [\nabla \vec{u} + (\nabla \vec{u})^T] : \nabla \vec{u} \quad (5)$$

where k is the turbulent kinetic energy and ε the dissipation.

As recommended by Launder and Spalding [12] and proved later by experiments, the adjustable constants appearing in Eqs. (2) to (5) have the following values: $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $C_\mu = 0.09$, $\sigma_K = 1.0$ and $\sigma_\varepsilon = 1.3$.

2.2 Boundary conditions

The set of equations presented previously provides a complete model for the description of turbulent flows. However, it cannot be easily solved as the presence of the non-linear convective term, first term in Eq. (1), creates a wide range of time and length scales [13]. Moreover, the assumptions used to derive the k- ε model are not valid close to walls where the tangential fluid flow velocity vanishes. Adequate description of the boundary conditions is then needed.

• Inlet conditions

For neutral stratification conditions, Richard and Hoxey (1993) [14] proposed for the incoming wind flow a logarithmic profile of mean velocity, and the following turbulent kinetic energy and dissipation rate:

$$u(y) = \frac{u_*}{\kappa} \left(\frac{y + z_0}{z_0} \right) \quad (6)$$

$$k(y) = \frac{u_*^2}{\sqrt{C_\mu}} \quad (7)$$

$$\varepsilon(y) = \frac{u_*^3}{\kappa y} \quad (8)$$

where y is the ascendant vertical coordinate, u_* is the friction velocity, κ the von Karman's constant and z_0 is the surface roughness. The chosen value for the turbulence model is

$$\kappa = \sqrt{(C_{\varepsilon 2} - C_{\varepsilon 1}) \sigma_\varepsilon \sqrt{C_\mu}} = 0.4237 \quad (9)$$

As an example, parameters adopted in CFD simulation as measured in the wind tunnel RUSHIL H3 experiment are given in Table 1.

Parameter	u_* [m/s]	u_∞ [m/s]	z_0 [mm]
Value (RUSHIL H3experiment)	0.178 m/s	4 m/s	0.16 mm

Table 1: Turbulence model constants associated to the inlet boundary condition

- **Ground condition**

For a turbulent flow, the flow close to a solid wall is very different from that occurring in the free stream condition. This means that the assumptions used to derive the k- ε model are not strictly valid close to walls. Theoretically, it is quite possible to modify the k- ε model so that it can describe more adequately the flow in wall regions. But, this is not always desirable because of the very high resolution requirement that results in that case. Instead, approximate analytical expressions are used to describe the flow at the walls. These expressions are known as wall functions.

The wall functions in COMSOL Multiphysics [10] are such that computational domain is assumed to be located at a distance δ_w from the wall, see Fig. 1.

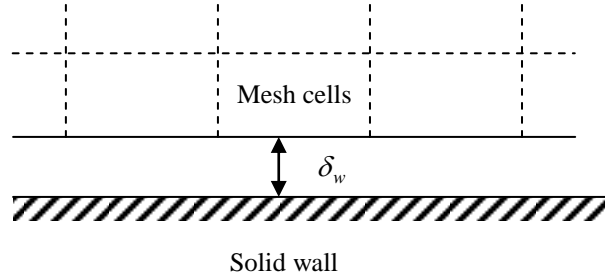


Fig. 1: Distance from wall to computational domain.

The distance δ_w is automatically computed so that $\delta_w^+ = u_\tau \frac{\delta_w}{\nu}$, where $u_\tau = C_\mu^{1/4} \sqrt{k}$ the friction velocity, becomes 11.06. This corresponds to the distance from the wall where the logarithmic layer meets the viscous sublayer if the buffer layer is discarded. δ_w is limited from below so that it never becomes smaller than half of the height of the boundary mesh cell. This means that u_τ can become higher than 11.06 if the mesh is relatively coarse.

The boundary condition for the velocity is a no-penetration condition which writes

$$\bar{\vec{u}} \cdot \vec{n} = 0 \quad (10)$$

and a shear stress condition

$$\vec{n} \cdot \bar{\vec{\sigma}} - (\vec{n} \cdot \bar{\vec{\sigma}} \cdot \vec{n}) \vec{n} = -v_\tau \frac{\bar{\vec{u}}}{\|\bar{\vec{u}}\|} \max(C_\mu^{1/4} \sqrt{k}, v_\tau) \quad (11)$$

with

$$\bar{\vec{\sigma}} = \nu (\vec{\nabla} \bar{\vec{u}} + (\nabla \bar{\vec{u}})^T) \quad (12)$$

$$v_\tau = \frac{\kappa \|\bar{\vec{u}}\|}{\ln(\delta_w^+) + \kappa B} \quad (13)$$

where $\kappa=0.41$ and $B=5.2$.

The turbulent kinetic energy is subjected to the following homogeneous Neumann condition:

$$\vec{n} \cdot \nabla k = 0 \quad (14)$$

The boundary condition for dissipation is

$$\varepsilon = \frac{C_\mu^{3/4} k^{3/2}}{\kappa \delta_w} \quad (15)$$

- **Outlet**

Flow is considered fully developed there, at the right side of the domain an outlet boundary condition is applied with the pressure fixed at the atmospheric pressure $p = p_{atm}$.

- **Top ground**

Dirichlet conditions for \bar{u} , k and ε , prescribing the values corresponding to the inlet profiles at the height of the computational domain. Additionally, one can also use slip conditions here: zero for the normal component of a vector, and zero gradient for tangential, zero gradient for any scalar.

2.3 Computational domain and grid generation

Considering the RUSHIL H3 experiment [11], the computational domain as obtained by a two-dimensional approximation of the problem has the configuration depicted in Fig. 2. The ground is assumed to be a horizontal plane and to present a hill like obstacle as shown in that figure. The domain of air surrounding the obstacle and a wind turbine has to be sufficiently large in order to contain the RUSHIL H3 hill and a standard wind turbine. It has also to justify model convergence with respect to truncation of the infinite ABL, that is to say enabling to simulate with enough precision air flow without causing significant perturbations resulting from the bounded domain structure. The dimensions of the RUSHIL H3 hill correspond to the height $h=117m$ and had shapes given parametrically by:

$$x = \frac{1}{2} \xi \left[1 + \frac{a^2}{\xi^2 + m^2 (a^2 - \xi^2)} \right] \quad |\xi| \leq a \quad (16)$$

$$y = \frac{1}{2} m \sqrt{a^2 - \xi^2} \left[1 - \frac{a^2}{\xi^2 + m^2 (a^2 - \xi^2)} \right]$$

where $m = n + \sqrt{n^2 + 1}$ and $n = h/a$ is the average slope.

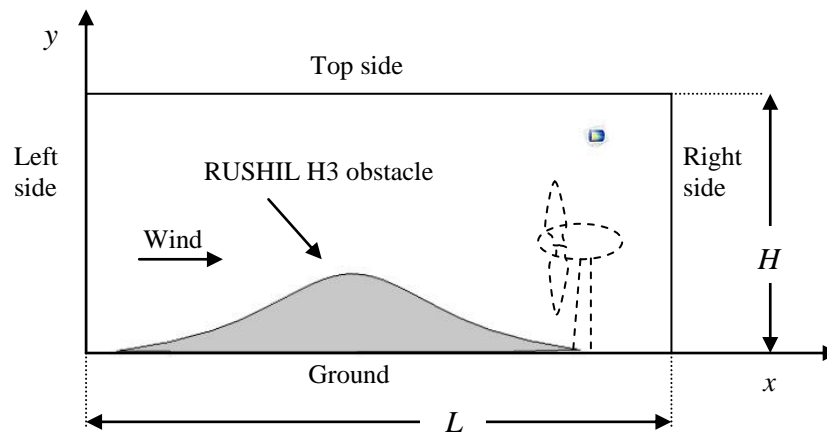


Fig. 2: Rectangular domain modeling the atmospheric air surrounding hill obstacle and a wind turbine

The domain of fluid to be modeled is part of the problem and has to be determined through an iterative procedure. It is fixed by the condition that at convergence the obtained solution should be stationary

and insensitive to the modification of the domain frontier location. It was found from posterior results that the adequate truncated domain of air can be modeled by a rectangle of length of $\pm 40h$ and height of $13.7h$.

CFD simulation is performed in the following by using COMSOL 5.1 Multiphysics software. At a practical level CFD simulation suffers in general from the uncertainty of creating an initial mesh that is susceptible to produce a sufficiently accurate solution for the actual problem and in particular for the required wall law resolution. Here also, a parametric study is performed in order to find an adequate mesh. The surface was meshed using free triangle elements. The complete mesh was definite with the maximum elements growth rate was fixed at 1.25. The mesh that yielded convergence is shown in Fig.3.

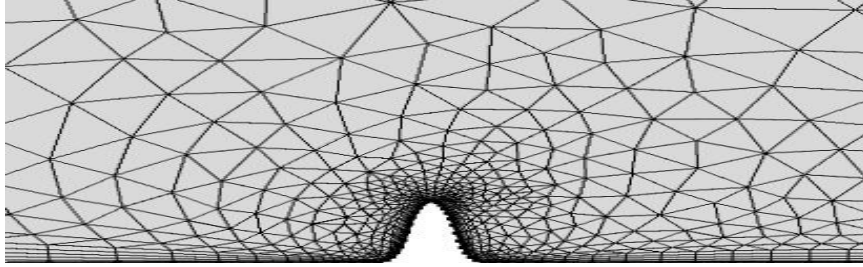


Fig. 3: Detail of the mesh elaborated with free triangle elements near the ground surface

In the following, the simulation is performed by using the computational domain introduced here above and with the mentioned mesh characteristics.

Fluid flowing inside the domain is air is considered with the following properties $\rho=1.225\text{kg/m}^3$ and kinematics viscosity $\nu=1.461\times 10^{-5}\text{kg/m/s}$.

Simulations results are verified by two important checks [15]:

- (1) investigate the solution to check that δ_w^- is small compared to the dimension of the geometry;
- (2) Also check that $\delta_w^+ = 11.06$ on most of the wall.

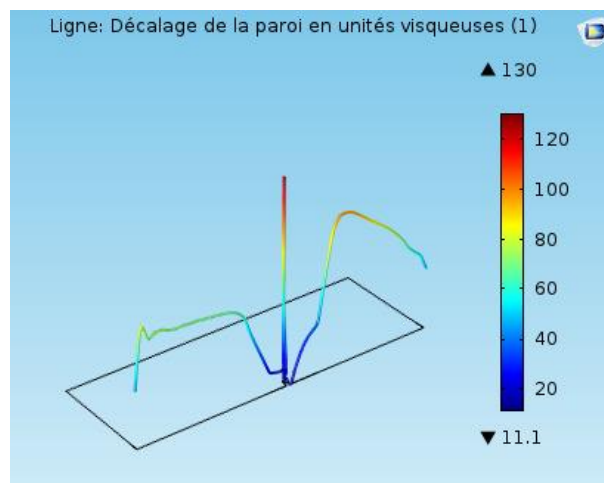


Fig. 4: The wall lift-off in viscous units, δ_w^+

If δ_w^+ is much higher over a significant part of the walls, the accuracy might become compromised. Both the wall lift-off δ_w and the wall lift-off in viscous units, δ_w^+ , are available as results and analysis variables under COMSOL Multiphysics software.

Fig. 4 gives the wall lift-off in viscous units, δ_w^+ as obtained through a simulation performed under COMSOL Multiphysics software. One can see that the required condition for adequacy $\delta_w^+ = 11.06$ is fulfilled here.

3 Results and discussions

The common computational CFD codes permit to easily represent the wind flow behavior for problems having a fairly homogenous configuration. In a case where the variation of the domain frontier is complex, with the presence of singularities that disturb highly the flow characteristics, the numerical resolution of the flow at the first element beyond that boundary is very hard to achieve. The limit between the two situations cannot be anticipated for a new problem.

The model hills in the RUSHIL study were, according to reference [11], of moderate slope. However, this was sufficient to exhibit marked flow separation on the lee slope. Considering the RUSHIL H3, Figs. 5 and 6 give the simulation results in terms of vertical profiles of the horizontal component of wind velocity at three different transversal sections: upwind base of domain, summit of domain and downwind base of domain. The experimental data as well as the simulation results provided by FLUENT software packages are also shown.

Fig. 7 gives the spatial distribution of the velocity field in terms of wind speed magnitude. Fig. 8 gives the spatial variation of the kinetic turbulent energy over hill.

From Figs. 5 and 6, one can notice that the wind speed profiles as obtained by simulation performed under COMSOL Multiphysics software are in good agreement with those obtained experimentally for all the three considered transversal sections. The results show that wind speed increase is observed at the hill summit when compared to a uniform flow over a flat ground. On the contrary, wind speed decay is observed at the downstream hill transversal section. This follows reflecting recirculation zones with a velocity feedback occurring in the negative direction as shown in Figs. 7.

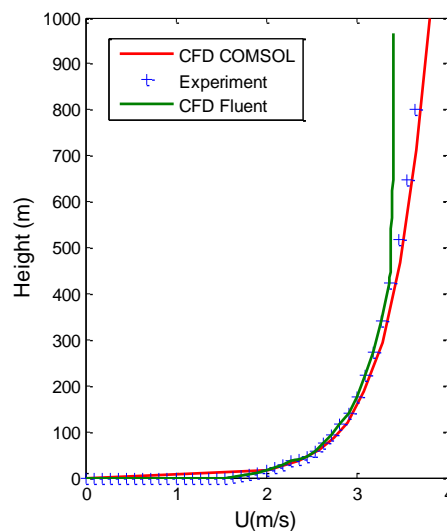


Fig. 5: RUSHIL H3, comparison of air flow velocity profiles at upwind base of domain

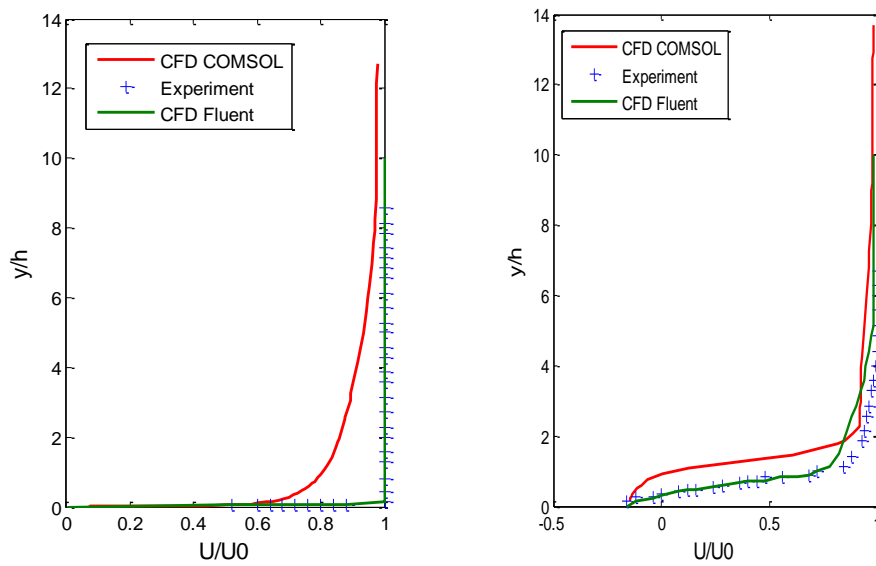


Fig. 6: RUSHIL H3, comparison of air flow velocity profiles;
(left) summit of hill (right) at downwind base

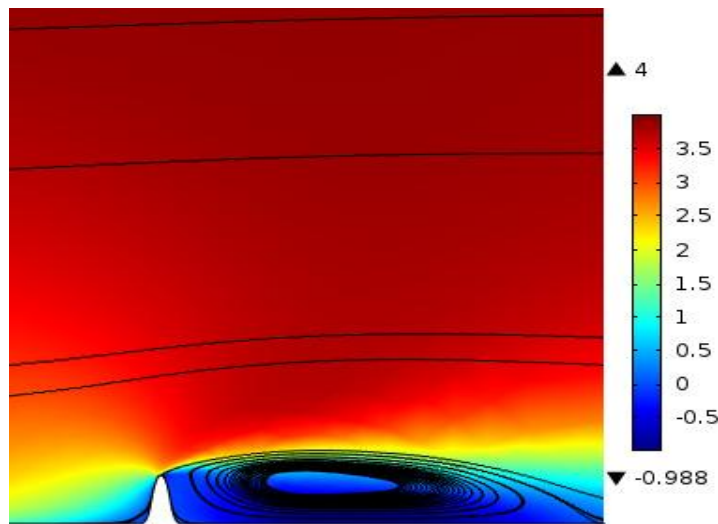


Fig. 7: Spatial variation of the wind speed velocity component x, and the mean separation streamline over hill.

Fig. 7 shows that the wind flow is completely differed from its initial pattern constituting a non-homogenous flow in speed and direction terms. An increase of speeds appears in the hill summit, while a deceleration of wind speed occurs at downstream. A boundary layer separation is also present. The obtained results in the actual simulation show that the presence of hill-like obstacle modifies considerably air flow pattern. Moreover, considering practical implementation issues of wind turbines, this occurs for heights that are habitually used to hold the rotor plane of these installations. Therefore, the presence of an obstacle is expected to modify largely the power pattern contained in wind. Consequently wind potential that can be extracted by a wind turbine is also dramatically changed. For this reason, wind turbines are usually mounted on top of hills.

Results of various simulations conducted parametrically as function of the ground roughness have shown that this parameter has a drastic influence on the wind speed field. For small values of roughness, the separation point was not observed.

CFD based simulation by using COMSOL Multiphysics software was found here to reproduce correctly the experimental results [11]; with the presence of a separation point and a recirculation pattern in the downhill zone. It can then be used to predict the wind flow pattern in the vicinity of irregular terrain. This can in particular help in making decision about best choice of wind turbine implementation site.

4 Conclusions

In this work, two-dimensional numerical simulation of flow pattern around a hill-like obstacle was conducted. Under neutral atmospheric stratification conditions, using the CFD approach where the turbulence was modeled by the $k-\varepsilon$ model, simulation of the air flow problem in the atmospheric boundary layer was achieved by using COMSOL Multiphysics software, which is based on the finite element method. For a more realistic representation of the flow field, a set of enhanced inlet conditions were used to satisfy the $k-\varepsilon$ turbulence local equations. A modified wall function formulation consistent with the inlet profiles of the neutral atmospheric flow was also applied at the ground surface. The proposed method was tested by performing comparison between the obtained numerical predictions with the experimental data measurements from RUSHIL H3 test. The obtained results indicated the capability of CFD based modeling to provide adequate description of the turbulent air flow over hill-like obstacle with moderate slope. The ground effects associated to this kind of obstacles were demonstrated with good enough accuracy. The wind turbine position could then be optimized in order to maximize wind energy conversion while minimizing turbulence to avoid excessive fatigue of blades and poor quality of produced electrical energy.

References

- [1] J.H. Ferziger, M. Peric, Computational Methods for Fluid Dynamics, 3rd edition, Springer-Verlag, Berlin, Germany, 2002.
- [2] A.D. Young, Boundary layers, BSP Professional Books, Oxford, pp. 180-182, 1989.
- [3] P. Carpenter, N. Locke, Investigation of wind speeds over multiple two-dimensional hills, Journal of Wind energy and Industrial Aerodynamics, 83 (1999) 109-120.
- [4] A.D. Ferreira, M.C.G. Silva, D.X. Viegas, A.M.G. Lopes, Wind tunnel simulation of the flow around two dimensional hills, Journal of Wind energy and Industrial Aerodynamics, 38 (1991) 109-122.
- [5] D.A. Paterson, J.D. Holmes, Computation of wind flow over topography, Journal of Wind energy and Industrial Aerodynamics, 46-47 (1993) 471-476.
- [6] G. Petersen, Wind tunnel modelling of atmospheric boundary layer flow over hills, PhD Thesis, Hamburg University, 2013.
- [7] H.G. Kim, C.M. Lee, H.C. Lim, N.H. Kyong, An experimental and numerical study on the flow over two dimensional hills, Journal of Wind energy and Industrial Aerodynamics 66 (1997) 17-33.
- [8] H.D. Nedjari, O. Guerri, M. Saighi, CFD Wind turbines wake assessment in complex topography, Energy conversion and management, 138 (2017) 224-236.
- [9] D. Kuzmin. A Guide to Numerical Methods for Transport Equations. University Erlangen-Nuremberg, 2010.

-
- [10] Anounymous, Introduction to Comsol Multiphysics, Version 5.1 Comsol, Sweden, 2015.
- [11] D.D. Apsley, Numerical Modelling of Neutral and Stably Stratified Flow and Dispersion In Complex Terrain, PhD Thesis, University of Surrey, UK, 1995.
- [12] B.E. Launder, D.B. Spalding, The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering, 3 (1974) 269-289.
- [13] A. Crespo, F. Manuel, J.C. Grau, J. Hernandez, Modelization of wind farms in complex terrain, Application of the Monteahumada wind farm, Proceedings of European Community Wind Energy Conference, Travemunde, pp. 440-443, 1993.
- [14] P.J. Richards, R.P. Hoxey, Appropriate boundary conditions for computational wind engineering models using the k- ϵ turbulence model, Journal of Wind Engineering and Industrial Aerodynamics, 46 & 47 (1993) 145-153.
- [15] D. Kuzmin, O. Mierka, S. Turek, On the Implementation of the k- ϵ Turbulence Model in Incompressible Flow Solvers Based on a Finite Element Discretization, International Journal of Computing Science and Mathematics, 1 (2007) 193-206.